High Resolution FE Model of Bone - Mesh

Author: Rajesh Bhaskaran, Cornell University
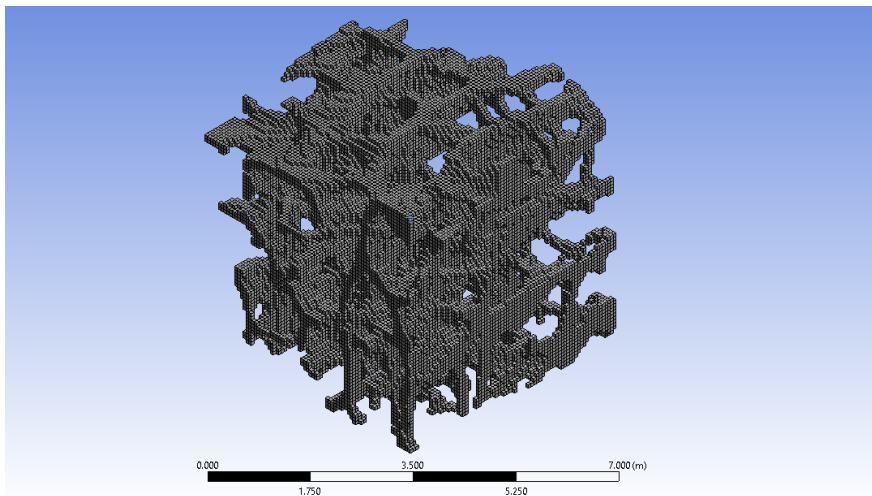
Mesh

Create the mesh

Once the External Model is updated, double click on Model in **Static Structural** to launch ANSYS Mechanical.

By using External Model to import the Abaqus file, a Geometry and Mesh setup are imported into ANSYS Mechanical. To verify the file is imported correctly, click on **Mesh** > **Statistics**, and verify that Nodes and Elements match the results listed below

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules